EFFECT OF BLANK HOLDER FORCE AND COUNTERFORCE ON WORKPIECE QUALITY IN SHEET METAL EXTRUSION PROCESS BY FINITE ELEMENT METHOD

KORAKOD PANGILILAD¹ & BOPIT BUBPHACHOT²*

¹Engineering, Division of Building and Grounds, Office of the President, Mahasarakham University, Maha Sarakham, Thailand

²Associate Professor, Heat-Pipe and Thermal Tools Design Research Unit (HTDR), Department of Mechanical Engineering, Faculty of Engineering, Mahasarakham University, Maha Sarakham, Thailand

ABSTRACT

This research is a study of the effect of blank holder force and counterforce on workpiece quality in the sheet metal extrusion process by finite element computer simulation. The workpieces in finite element simulation were medium carbon steel (AISI: 1045) with a thickness of 10 mm. The simulation model requires that a punch diameter of 12.7 mm, a die diameter of 5 mm, different blank holder force and counterforce with the changing material properties were compared at the depth of 1, 2, 3 and 4 mm respectively. From the results, it was found that the increased blank holder force and counterforce affect a small increase in effective stress and effective strain in workpieces. The results of the study show the influence of the blank holder force and counterforce affected the quality of workpiece in the sheet metal compression process.

KEYWORDS: Blank holder force; Counterforce; Sheet Metal Extrusion Process & Finite Element Method

Received: Nov 02, 2020; Accepted: Nov 22, 2020; Published: Dec 22, 2020; Paper Id.: IJMPERDDEC202021

1. INTRODUCTION

The sheet metal press process is the most important process in the automotive parts manufacturing industry. In order to produce a quality product, it is necessary to study the parameters for use as a basis for machining molds which can study various variables by machining molds and then doing a real production experiment. In this case, there is used a high budget for studying, but there is another way to save money and time to experiment. This method is a computer simulation. The computer simulation is being developed and becoming more and more popular. Since the computer has developed to assist in the mathematical processing by finite element method which makes it convenient to analyzed engineering problems. The finite element computer simulation is a very useful method for product design, planning production processes, helps in selecting suitable variables in the mold design and determining the optimal forming process. In addition, it can also be used to analyze production problems to determine the fault cause. This method will reduce the number of trial and error and save time and cost for the experiment. Therefore, researchers have studied and submitted research articles that confirm the accuracy of the actual process experiments compared with the process simulation using the finite element method. In 2010, Suriyapha et al [1] studies the effect of die radius on the sheet metal extrusion in the fine blanking process (SME-FB) surface which investigated the formation of the failure deflection with respect to the several die radiiuses by using the Finite Element Method (FEM). From the results, it indicated that applying the small die radius caused the
material flow difficulty resulting in a decrease of smooth surface. Vice versa, in the case of a large die radius, the material flow easy is resulting in the increase of smooth surface. The FEM simulation results of a larger die radius will cause residual stress at the workpiece. In 2011, Suriyapha et al [2] studies explain the characteristic of material-flow behavior on SME-FB. That is the cause of the surface crack and end-rod shrinkage which are the general problems in the SME-FB. In this study, the medium steel S45C (JIS) was used as a material extruded vary in punch and die radius of 0.0, 0.1, 0.2 and 0.3 mm. Therefore, the material-flow behavior on the SME-FB process has investigated the formation of the defection with respect to the several die radius by using the Finite Element Method (FEM). From the results, it indicated that applying the small die radius caused the material flow difficulty resulting in a decrease of smooth surface. In contrast, in the case of a large die radius, the material can flow easily is resulting in the increase of smooth surface. Moreover, the FEM simulation results of a larger die radius will cause residual stress at the workpiece. In 2015, Suriyapha et al [3] investigated the characteristics of the material-flow behavior during the formation and its effect on the microstructure of the extruded sheet metal using the FEM. The actual parts and FEM simulation model have used a blank material made from AISI-1045 steel with a thickness of 5 mm; the material's behavior was determined to subject the punch penetration depths of 20%, 40%, 60%, and 80% of the sheet thickness. The results indicated the material behavior was characterized by large defects on the extrusion shrinkage, extrusion buckling and extrusion die roll appearance as shown in Figure 1. Therefore, this research is a study of the effect of blank holder force and counterforce on workpiece quality in the SME process by finite element computer simulation to help with such problems and basic data before mold machining.

![Figure 1: Results (a) cross-section of workpiece, (b) Experiment and (c) FEM [3]](image)

2. MATERIALS AND METHODS

2.1 Material properties

The FEM can be a simulation in which mathematics computing in mechanical properties equations of material. The medium carbon steel (AISI:1045) was used as a stock material model, mechanical properties were determined by tension
testing as shown in Figure 2. The FEM results of the tensile test show that the elongation was 26% and the tensile strength was 530 MPa [3]. The friction coefficient for the FEM simulation was obtained by employing the ring compression test, and it was 0.04 [4, 5]. Finally, the FEM results were analytical as geometry and effective stress value.

![Figure 2: Comparison of Tensile Test Between Experimental and FEM Simulation Results](image)

2.2 FEM Simulation Method

The modeling of the sheet metal extrusion process is significant being the ability to model the part as axisymmetric. In this case, only half of the part must be modeled, with the nodes of workpiece on its axis of symmetry as shown in Figure 3. Therefore, the material specimen employed in this study was AISI-1045 steel with a sheet thickness of 5 mm and blank diameter of 38 mm. The punch and die diameters were 11.27 mm and 10.00 mm, respectively, and were formed via a SME process with extrusion ratio $R = 1.27$. The blank holder force was 3 level as 16, 32 and 48kN. The counterforce was 2 level as 500N and 32kN.

One can see the concentration of elements in the area where the shear zone will build up. The finite element method (commercial code DEFORM-2D) was used for the FEM simulation. The shape of the elements of the blanked material was a rectangular element (4-noded square element type). It was also noted that approximately 5,000 elements were designed for the blanked material. Calculations were performed by remeshing so that the divergence of the calculations due to excessive deformation of the elements was prevented. The FEM simulation conditions are shown in Table 1.
Medium carbon steel (AISI-1045) was used as a blank material and a flow curve equation was determined by tensile testing experiments and its mechanical properties are shown in Table 1. The constitutive equation was calculated from the tensile testing results. Therefore, the strength coefficient value 850 and the strain-hardening exponent value 0.478 were obtained. In order to investigate the fundamental failure and the form of the blank surface by means of the FEM during blanking, the fracture criterion equation and critical fracture value were considered. In this study, Oyane’s ductile fracture criterion equations were selected as $0.157$ [6], which were investigated and also usually used for the sheet metal extrusion process. A critical fracture value was determined in each fracture criterion equation by using a tensile testing experiment. A critical fracture value was used which agree well with tensile strength and elongation between the FEM simulation results of tensile testing. In this study, the friction coefficient of 0.12 was used for the blanking process.

### Table 1: FEM Simulation Conditions.

<table>
<thead>
<tr>
<th>Simulation model</th>
<th>Axisymmetric model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blank material [3]</td>
<td>AISI-1045 $\varnothing = 38$ mm, $t = 5$ mm $\sigma_B = 530$N/mm$^2$ and $\varepsilon = 26%$</td>
</tr>
<tr>
<td>Punch penetration</td>
<td>$P_{\text{depth}} = 1, 2, 3$ and $4$ mm</td>
</tr>
<tr>
<td>Punch [3]</td>
<td>AISI-M4 $\varnothing_p = 12.70$ mm and $R_p = 0.20$ mm</td>
</tr>
<tr>
<td>Die [3]</td>
<td>AISI-M4 $\varnothing_d = 10.00$ mm and $R_d = 0.20$ mm</td>
</tr>
<tr>
<td>Blank holder force</td>
<td>AISI-M4, $F_B = 16, 32$ and $48$ kN</td>
</tr>
<tr>
<td>Counterforce</td>
<td>AISI M4, $F_C = 500$ N and $32$ kN</td>
</tr>
<tr>
<td>Flow curve equation [3]</td>
<td>$\sigma = 850\varepsilon^{0.478} + 385$</td>
</tr>
<tr>
<td>Fracture criterion equation [6]</td>
<td>Oyane (constant $\alpha = 1$, critical fracture value $C = 0.157$)</td>
</tr>
<tr>
<td>Friction coefficient</td>
<td>$\mu = 0.04$</td>
</tr>
</tbody>
</table>

### 3. RESULTS AND DISCUSSIONS

#### 3.1 The FEM Simulation

In this study, the experimental result confirmed that the geometry measurement values between the actual forming [3] and the FEM simulation results were the same as shown in Figure 4. The earliest stage of the FEM simulation is shown in Figure 4(a). The workpiece protruded through the die orifice due to the punch thrust. The elements were a little flat near the punch edge and the workpiece compression reaction as the compression stress most appeared around the punch edge. In
contrast, the elements nearby the die edge were pushed forward easily and tension stress was mostly present in such an area. Figure 4(b) shows when the thrust punch penetrated 2 mm. The elements were increasingly flat around the punch and die edges. The compression stress was intense around the punch edge and suffused between the punch and die edges. The thrust punch penetrated 3 mm as shown in Figure 4(c). The elements were increasingly flat and pervaded around the punch and die edges. The compression stress was intense and expanded in the workpiece. The compressive stress was intense around the punch edge and there was increased convergence between the punch and die edges. The final state of the FEM simulation was a thrust punch that penetrated 4 mm. The elements were extremely fine and pervaded the punch and die surface and the compression stress intensity was greater between the punch and die surface. The compression stress density increased around the punch edge and there was convergence condensation between the punch and die edge as shown in Figure 4(d). Therefore, the extrusion shrinkage, the extrusion die roll and buckling were increased with the depth of punch penetration as shown in Figure 4 (a) punch penetration as 1 mm, (b) punch penetration as 2 mm, (c) punch penetration as 3 mm and (d) punch penetration as 4 mm.

![Figure 4: The FEM Simulation Results of F_b=16kN and F_c=500N](image)

### 3.2 Effect of Blank Holder Force and counterforce on Extrusion Defections

From the study, the condition of blank holder force values were 3 levels as 16, 32 and 48 kN. It was found that punch penetration at 3 mm, the workpiece occurs extrusion shrinkage but the extrusion die roll and extrusion buckling value
decreased with blank holder force increased as shown in Figure 5. Therefore, Figure 6 shows the result of the blank holder force and counterforce value of 32 kN. It was found that punch penetration as 1, 2, 3 and 4 mm, the workpiece has not occurred extrusion shrinkage with an increase of the blank holder force and counterforce value of 32 kN.

Figure 5: The FEM Simulation Results of $F_c = 500$N for Punch Penetration as 3 mm

Figure 6: The FEM Simulation Results of $F_B = 32$kN and $F_C = 32$N
CONCLUSIONS

This research is a study of the effect of blank holder force and counterforce on workpiece quality in the sheet metal extrusion process by finite element computer simulation to help with such problems and basic data before mold machining. From the present study, the conclusions can be summarized as follows:

- The increasing of blank hold force and counterforce value was decreased the extrusion defects.
- The increased blank holder force and counterforce value affect a small increase in effective stress value in workpieces.

ACKNOWLEDGEMENTS

This research was financially supported by the Faculty of Engineering, Mahasarakham University (Grant year 2021). The authors would like to thank the Heat-Pipe and Thermal Tools Design Research Unit (HTDR), Faculty of Engineering, Mahasarakham University, Maha Sarakham, Thailand for equipment and tools support and Department of Tool and Material Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Bangkok, Thailand for the finite element program (DEFORM-2D) support.

REFERENCES

AUTHOR’S PROFILE

Mr. Korakod Panglilad received his B.Eng. degree in Practice Engineering from Mahasarakham University, Thailand, in 2020. He is currently a Head of Maintenance Group at Division of Building and Grounds, Office of the President, Mahasarakham University, Maha Sarakham, Thailand. His research interests include Metal Forming (Sheet Metal Extrusion Process), FEM Simulation in Metal Forming Process.

Mr. Bopit Bubphachot received his Ph.D. degree in Engineering Mechanics and Energy from the University of Tsukuba, Japan, in 2008. He is currently an Associate Professor at the Faculty of Engineering at Mahasarakham University in Thailand. His research interests include Metal Forming (Fine Blanking Process), FEM Simulation in Metal Forming Process, Fatigue and Creep-Fatigue strength for Perforated Plate, Heat pipe, and Heat exchanger.